FEAMAC Example 1

***Description*:**

This is a simple FEAMAC problem involving two unit 3D elements with a prescribed displacement of 0.03 in. in the 1-direction. Both elements are assigned the same MAC/GMC material, a 33% volume fraction SiC/Ti composite with a simple 2×2 GMC repeating unit cell (see Fig. 2), thus all eight integration points in both elements have their material constitutive response described by MAC/GMC. The viscoplastic material response of the Ti is modeled via the GVIPS constitutive model within MAC/GMC [2]. Because the model is 2 in. long in the applied loading direction, the applied displacement corresponds to an applied strain of 0.015. The FEAMAC problem is thus identical to a stand-alone MAC/GMC problem in which a strain of 0.015 is applied to the composite. A constant temperature of 23 °C is employed and the loading is applied over 150 seconds for a global strain rate of 10^{-4}/second. Via commenting and un-commenting several lines in the ABAQUS and MAC/GMC input files, both longitudinal loading (in the fiber direction) and transverse loading (normal to the fiber direction) are considered in separate executions.

Required Files:

The following files should be placed in the ABAQUS working directory:

File	**Purpose**
`feamac_ex1.inp`	ABAQUS input file
`SiC-Ti_33.mac`	MAC/GMC input file describing the SiC/Ti composite material
`feamac.for`	User-defined subroutines for FEAMAC

Because this is the first FEAMAC example problem, both the ABAQUS and MAC/GMC input files are given in their entirety below.

<u>`feamac_ex1.inp`</u>

```
*HEADING
FEAMAC Example Problem 1
*NODE,NSET=ALL
1,0.0,0.0,0.0
2,0.0,1.0,0.0
3,0.0,1.0,1.0
4,0.0,0.0,1.0
5,1.0,0.0,0.0
6,1.0,1.0,0.0
7,1.0,1.0,1.0
8,1.0,0.0,1.0
9,2.0,0.0,0.0
10,2.0,1.0,0.0
11,2.0,1.0,1.0
12,2.0,0.0,1.0
*ORIENTATION, NAME=TRANS
0.0, 1.0, 0.0, -1.0, 0.0, 0.0
*ORIENTATION, NAME=LONG
1.0, 0.0, 0.0, 0.0, 1.0, 0.0
*ELEMENT, TYPE=C3D8, ELSET=EL1
1,1,2,3,4,5,6,7,8
*ELEMENT, TYPE=C3D8, ELSET=EL2
2,5,6,7,8,9,10,11,12
*NSET, NSET=ALL
1,2,3,4,5,6,7,8,9,10,11,12
*ELSET, ELSET=EALL
EL1, EL2
**---------------------------------------------------------------------
** Comment and un-comment the following lines to switch between
** longitudinal and transverse loading
```

```
*SOLID SECTION, MATERIAL=SiC-Ti_33.MAC, ELSET=EALL, ORIENTATION=LONG
 1.0
***SOLID SECTION, MATERIAL=SiC-Ti_33.MAC, ELSET=EALL, ORIENTATION=TRANS
** 1.0
**----------------------------------------------------------------------
*MATERIAL,NAME=SiC-Ti_33.MAC
*USER MATERIAL, CONSTANTS=0
*DEPVAR
207
**----------------------------------------------------------------------
*INITIAL CONDITIONS, TYPE=TEMPERATURE
ALL, 23.0
*INITIAL CONDITIONS, TYPE=SOLUTION, USER
**----------------------------------------------------------------------
*STEP, AMPLITUDE=RAMP, INC=2000000
STEP ONE - LOAD UP
*STATIC, DIRECT
2.0,150.0
*BOUNDARY
1,1,3
2,1
3,1
4,1
9,1,1,0.03
10,1,1,0.03
11,1,1,0.03
12,1,1,0.03
*TEMPERATURE
ALL, 23.0
*OUTPUT, FIELD, FREQUENCY=1
*ELEMENT OUTPUT, ELSET=EALL
SDV
S
E
THE
*NODE OUTPUT, NSET=ALL
U
*OUTPUT, HISTORY, FREQUENCY=1
*ELEMENT OUTPUT, ELSET=EALL
S11
E11
*NODE OUTPUT, NSET=ALL
U
*ENDSTEP
**----------------------------------------------------------------------
```

In order to execute the longitudinal and transverse loading cases, the two `*SOLID SECTION` specifications should be commented and un-commented. The MAC/GMC input file name has been specified as the material name with `*MATERIAL,NAME=SiC-Ti_33.MAC`, and the required `*USER MATERIAL, CONSTANTS=0` specification has been made. The `*DEPVAR` specification has been made with a value of $\text{DEPVAR} = 39 + 42\, N_\alpha\, N_\beta\, N_\gamma \left(N_{\text{int}}\right)^2 = 39 + 42 \times 1 \times 2 \times 2 \times \left(1\right)^2 = 207$. The required `*INITIAL CONDITIONS, TYPE=SOLUTION, USER` specification has been made, as has the initial temperature specification, indicating a starting temperature of 23 °C. The single step of the applied loading is defined between the `*STEP` and `*ENDSTEP` specifications. Under `*STATIC, DIRECT` a step time period of 150 seconds with a time increment of 2 seconds is specified. The boundary conditions pin node 1 and restrain the base face nodes (1 – 4) against displacement in the applied loading (1) direction. The opposite face nodes (9 – 12) are prescribed a displacement of 0.03 in, which is applied linearly over the 150 second step time period. The step ending temperature for all nodes is set to the same value as the initial temperature (23 °C) using the `*TEMPERATURE` specification. Field and history output for the ABAQUS output database are specified using `*OUTPUT, FIELD` and `*OUTPUT, HISTORY` in conjunction with the `*ELEMENT OUTPUT` and `*NODE OUTPUT` specifiers.

SiC-Ti_33.mac

```
33% SiC/Ti, 2x2 RUC
*CONSTITUENTS
  NMATS=2
  M=1 CMOD=6 MATID=E
  M=2 CMOD=4 MATID=A
*RUC
  MOD=2 ARCHID=11 VF=0.33 R=1. F=1 M=2
*XYPLOT
  FREQ=1
  MACRO=1
# -- Comment and un-comment the following lines
#    to switch longitudinal and transverse output
   NAME=SiC-Ti_33 X=1 Y=7
#   NAME=SiC-Ti_33 X=3 Y=9
  MICRO=0
*END
```

The MAC/GMC input file `SiC-Ti_33.mac` is quite simple. The constituent materials and repeating unit cell are specified, along with the xy data output. As indicated, the two xy data output requests should be commented and un-commented to generate the desired output for the longitudinal and transverse cases. If no MAC/GMC xy data is desired, the `*XYPLOT` specification can be omitted.

Execution:

This problem can be executed via the following command at the ABAQUS command line:

```
abaqus –j feamac_ex1 –user feamac interactive
```

The `–j` specification indicates the job name (i.e., ABAQUS input file name), while the `–user` specification indicates the file containing the FEAMAC user-defined subroutines. The `interactive` specification provides detailed information on the problem execution during the execution and is optional.

Output:

The output for this problem is written to the ABAQUS output database file `feamac_ex1.odb` for post-processing in ABAQUS/CAE, ABAQUS/Viewer, or other appropriate finite element post-processing software. In addition, MAC/GMC output is written to 16 ASCII files named `SIC-TI_33_macro_EL`*i*`_PT`*j*`.data`, where *i* and *j* are the element number and integration point number in the ABAQUS finite element model, respectively. These files contain the macro (repeating unit cell) level output specified in the MAC/GMC input file. "`_macro`" and "`_EL`*i*`_PT`*j*`.data`" have been appended to the name specified for the macro output in the MAC/GMC input file. Finally, a MAC/GMC output file is written for each MAC/GMC input file employed in the ABAQUS model. In this case, this file is `SiC-Ti_33.out`. It contains an echo of the MAC/GMC input file data and results in the form of effective properties. If an error is found in the MAC/GMC input, a message describing the error will be written to this output file.

Results:

Results in the form of stress-strain curves are plotted in Fig. 3. The curves labeled "FEAMAC" are plotted from the ASCII output in the files `SIC-TI_33_macro_EL`*i*`_PT`*j*`.data`, which are all identical due to the uniform state of stress and strain that results in the model. As mentioned above, because the problem simulates uniform stress and strain throughout the model, results are identical to a strain-controlled stand-alone MAC/GMC problem on the SiC/Ti composite. This fact is illustrated in Fig. 3 as the results of stand-alone MAC/GMC analyses are plotted as well. As expected, the composite is stiffer and exhibits less inelastic strain in the longitudinal case compared to the transverse case. Figures 4 and 5 provide plots of the uniform σ_{11} stress and equivalent plastic strain fields that result in the model due to the applied loading in the case of longitudinal fiber orientation.

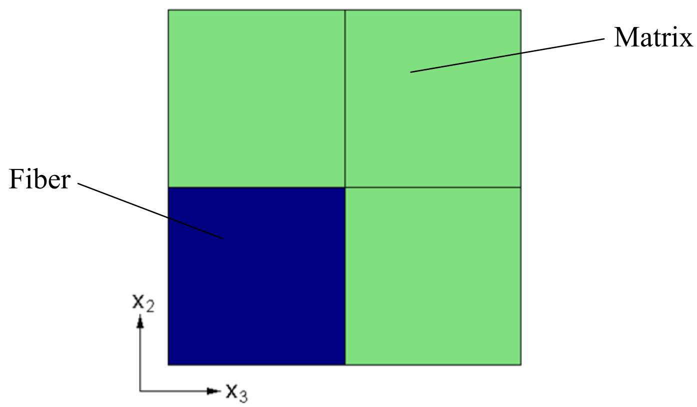

Fig. 2. 2×2 GMC repeating unit cell.

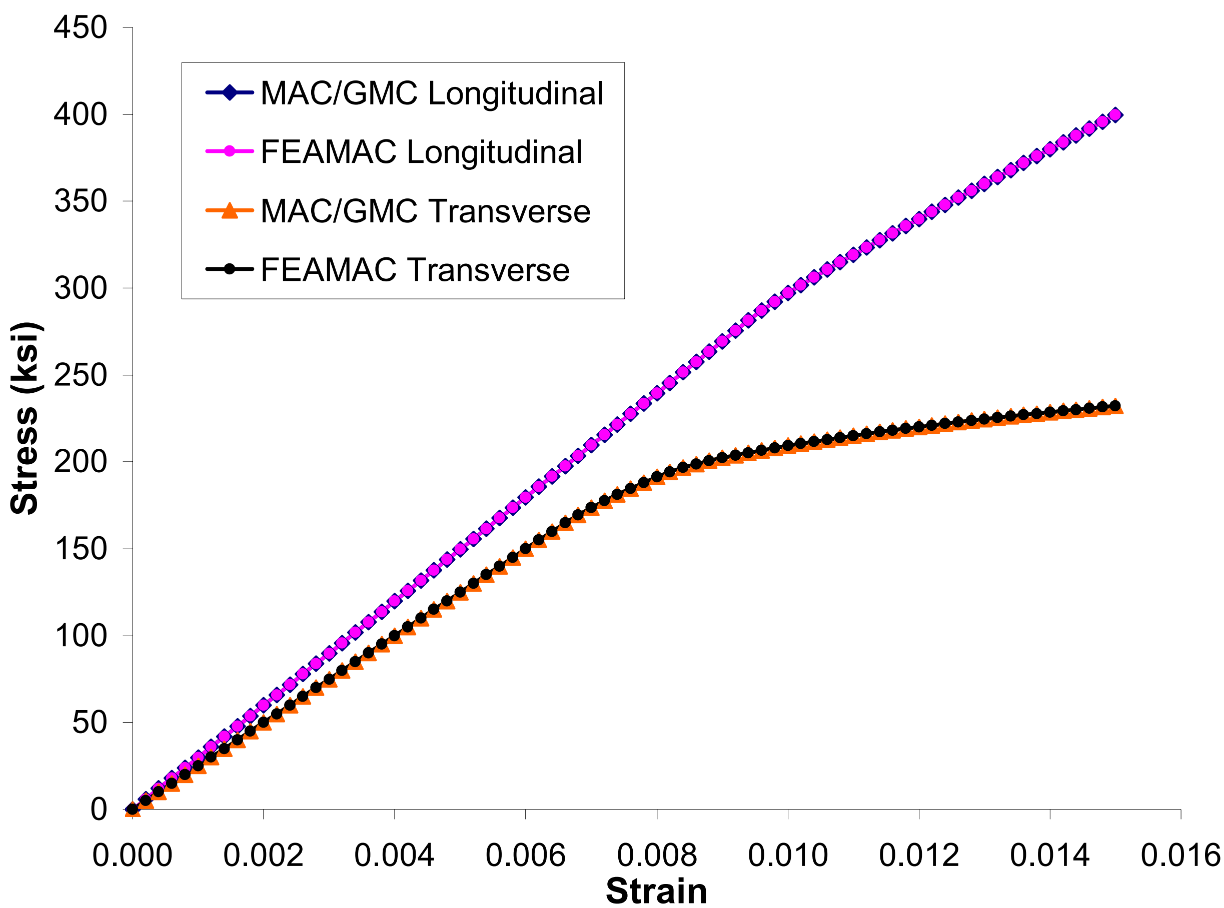

Fig. 3. Longitudinal and transverse stress-strain curves for a 33% SiC/Ti composite represented by a 2×2 GMC repeating unit cell predicted by stand-alone MAC/GMC and FEAMAC.

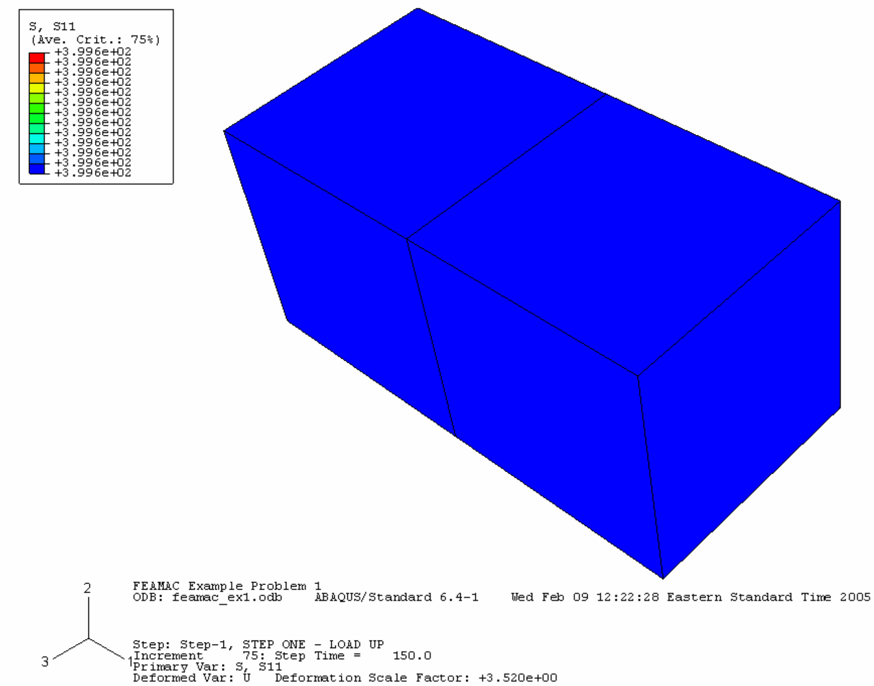

Fig. 4. σ_{11} stress field (ksi) at the end of the loading step in the 2 elements comprising the ABAQUS model as predicted by FEAMAC. The fiber orientation is longitudinal (in the 1-direction) in this case.

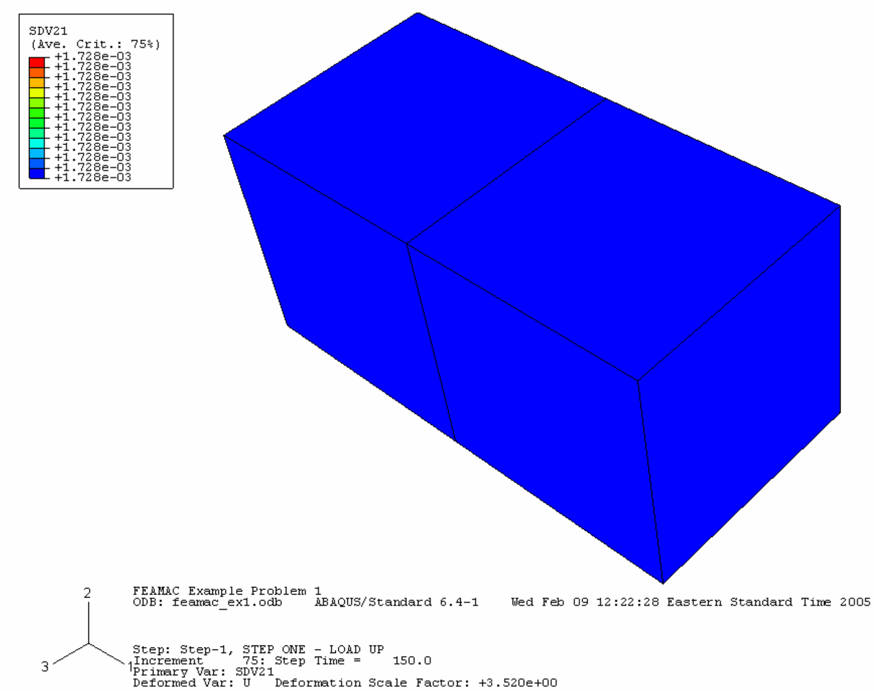

Fig. 5. Equivalent plastic strain field at the end of the loading step in the 2 elements comprising the ABAQUS model as predicted by FEAMAC. The fiber orientation is longitudinal (in the 1-direction) in this case.